MODELING OF FLOW CHARACTERISTICS IN A PUMP SUMP PHYSICAL MODEL USING COMPUTATIONAL FLUID DYNAMICS

MOHD REMY ROZAINY BIN MOHD ARIF ZAINOL

UNIVERSITI SAINS MALAYSIA

2007

MODELING OF FLOW CHARACTERISTICS IN A PUMP SUMP PHYSICAL MODEL USING COMPUTATIONAL FLUID DYNAMICS

by

MOHD REMY ROZAINY BIN MOHD ARIF ZAINOL

Thesis submitted in fulfilment of the requirements for the degree of Master of Science

OCTOBER 2007

MODELING OF FLOW CHARACTERISTICS IN A PUMP SUMP PHYSICAL MODEL USING COMPUTATIONAL FLUID DYNAMICS

by

MOHD REMY ROZAINY BIN MOHD ARIF ZAINOL

Thesis submitted in fulfilment of the requirements for the degree of Master of Science

OCTOBER 2007

ACKNOWLEDGEMENTS

In the name of Allah, The Compassionate, The Merciful

The first person I would like to thank is my principal supervisor Assoc. Prof. Dr. Hj. Ismail Abustan. His overly enthusiasm and integral view on research and his mission for providing knowledgeable and high quality student has made deep expression on me. I owe him lots of gratitude for having shown me this way of research. He could not realize how much I learned from him. Besides of being of excellent supervisor, he was also close like a father to me and other post-graduate students.

Special thanks also go to Assoc. Prof. Dr. Zulkifly Abdullah, as my co-supervisor for his extremely knowledgeable advices, tips and encouragement that help me a lot in staying on the right track for this research.

I owe also my sincere gratitude to Ir. Hj. Mubarak Hussein, the director of Mechanical Services Division of Drainage and Irrigation Department (DID) of Malaysia, who gave me opportunity to work with them and gave me untiring help during my laboratory test in their lab. I warmly thank to Hj. Zainuddin, Mr. Lim and all technical staff of Mechanical Services Division for their valuable expertise advised and friendly help within my experimental study and data collection.

My appreciation and thanks is also conveying to all the academic and technical staff of the School of Civil Engineering, Universiti Sains Malaysia for their assistance directly and indirectly throughout the duration of this work.

I would like to express my greatest gratitude and appreciation to the Intensity Research in Priority Area (IRPA) for its financial support during my study. Without this support I will not able to manage my financial properly.

I feel a deep sense of gratitude to my parents Mohd Arif Zainol B. Mohd Akib and Asmah Bt. Mat who formed part of my vision and taught me the good things that really matter in my life. To all my siblings, I'd really like to express my thanks, especially to my brothers Mohd Hussainy and Mohd Fizry and sister, Jurainy for just being there. I am also grateful to my lovely wife, Noor Hana Hanif Abu Bakar for being my source of inspiration for her endless support throughout the hard time. The good taught from her still provides a persistent inspiration for my journey in this life.

Last but not least, I wish to express my special thanks to all my fellow colleagues and friends for their invaluable ideas throughout this work.

Wassalam

TABLE OF CONTENTS

CHAPTER SIX : CONCLUSIONS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

ρ	Density
μ	Dynamic viscosity
υ	Kinematics viscosity
Θ	Swirl angle
ε	Turbulence dissipation rate
a	Inlet width
A	Cross-sectional area of the intake
A_{inlet}	Inlet cross-section
b	Inlet depth
D_1	Suction intake diameter (bell mouth)
D_2	Column intake diameter
d_v	Diameter of vortimeter vane
F_D	Froude number
F_m	Froude model parameter
F_p	Froude prototype parameter
F_r	Froude ratio parameter
g	Gravitational constant
H	Water level / depth
k	Turbulence kinetic energy
l	Characteristic length
L	Characteristic length depending on water level
Q	Flow through the pump intake
Q_{inlet}	Inlet flow rate
R	Number of revolutions of the vane per minute
Re	Reynolds number
S	Submergence depth
TI	Turbulence intensity

U	Inlet velocity
V	Velocity
V_A	Axial velocity
V_c	Cross-flow velocity
V_{ch}	Approach velocity
V_{inlet}	Inlet velocity
V_R	Rotational velocity
V_x	Pump bay velocity

LIST OF ABBREVIATION

ADV	Acoustic Doppler Velocimeter
CFD	Computational Fluid Dynamics
DID	Department of Irrigation and Drainage
PVC	Polyvinyl chloride
RANS	Reynolds-average Navier Stokes
TPFM	Thermo Polysonic Flow Meter
VVM	Valeport Velocity Meter
3D	Three dimensions

LIST OF APPENDICES

LIST OF PUBLICATIONS & SEMINARS

Mohd Remy Rozainy, M.A.Z., Abustan, I., Abdullah, M.Z., Muhad, R.M. Nawi., and M.A. Ismail. (2006). Computational Fluid Dynamic (CFD) to Modeling of Flow Characteristics in Physical Model of Pump Sump. 1st Civil Engineering Colloquium, School of Civil Engineering, Universiti Sains Malaysia.

PERMODELAN CIRI-CIRI ALIRAN DALAM MODEL FIZIKAL TAKUNGAN PAM MENGGUNAKAN PERKOMPUTERAN DINAMIK BENDALIR

ABSTRAK

Kajian ini dijalankan bagi memodelkan ciri-ciri aliran dalam model fizikal takungan pam menggunakan Dinamik Bendalir Berkomputer (CFD) melalui kod FLUENT™ 6.2. Prosedur atau tatacara ujikaji melibatkan cerapan data mengunakan meter halaju , meter aliran, dan meter pusaran (Rotometer / Vortimeter). Tiga jenis pengukuran diambil iaitu halaju, aliran, dan sudut pusaran untuk sembilan kajian kes yang telah dinilai pada tiga kedalaman air yang berbeza (0.3m, 0.24m, dan 0.18m), masing-masing pada tiga kadar alir yang berbeza (15L/s, 10L/s, dan 4L/s); Sejumlah 162 titik cerapan bagi setiap kes. Suatu ujian visual yang melibatkan teknik pengesanan penunjuk bewarna (dye) turut dijalankan untuk mencirikan aliran. Dalam kajian ini, pembangunan model CFD yang komprehensif telah digunakan dalam rekabentuk takungan pam. Perbandingan dengan kaedah ujikaji dan model CFD akan dibincangkan dengan lebih lanjut. Hasilan FLUENT™ menggambarkan ciri-ciri asas magnitud vektor-vektor halaju (m/s), kontur vektor halaju (m/s) dan kontur tekanan pegun (pascal). Persetujuan yang baik diperolehi antara keputusan simulasi dan ujikaji. Lokasi vortex dalam keputusan ujikaji hampir menyamai keputusan simulasi CFD yang diperolehi dalam kajian ini. Purata perbezaan halaju di antara ujikaji dan simulasi ialah 4.2% dan 11.6%. Sementara itu, nilai pekali regresi (R^2) yang berada dalam julat 0.98 ke 0.99 telah diperolehi untuk kehubungan plotan berselerak antara data ujikaji dan simulasi. Oleh itu, daripada kajian ini, kesimpulan yang boleh dibuat ialah CFD dapat digunakan untuk simulasi atau di masa akan datang menggantikan model fizikal takungan pam.

MODELING OF FLOW CHARACTERISTICS IN A PUMP SUMP PHYSICAL MODEL USING COMPUTATIONAL FLUID DYNAMICS

ABSTRACT

This study attempts to model the flow characteristic in a pump sump physical model by using Computational Fluid Dynamics (CFD) code FLUENT^{TM} 6.2. The experimental procedures include the data collection using a velocity meter, flow meter and swirl meter (Rotometer / Vortimeter). Three types of measurements were conducted which are velocity, flow, and swirl angle for nine cases which had been evaluated at three different water depths (0.3m, 0.24m and 0.18m) and at three different flow rates (15L/s, 10L/s and 4L/s); a total of 162 measurement points per case. A visual test that involves the dye tracing technique was also carried out to characterize the flow. Further, in this study, a comprehensive CFD model of pump bays was developed. The comparison of experimental method and CFD model is discussed in details. The FLUENT^{TM} outputs illustrate the basic features of magnitudes of velocity vectors (m/s), contour of velocity vector (m/s) and static pressure contour (pascal). A good agreement is determined between simulation and experimental results. The locations of the vortices in the experimental results closely match the CFD simulation results obtained. The average velocity magnitude difference between experimental and CFD simulated result is recorded at 4.2% to 11.6%. Moreover, the regression coefficient (R^2) values of velocity magnitude ranging from 0.98 to 0.99 were obtained from the scattered plot relationship between experimental and simulated data. Thus from the study, it can be concluded that the CFD can be used to simulate flow characteristics in pump sump as an alternative to physical modeling of pump sump.

CHAPTER 1
INTRODUCTION

1.1 Background of the Study

The flow phenomena that arise in the pump bays of water intake structures have been studied experimentally by many researchers (Dicmas, 1987; Larsen and Padmanabhan, 2001; Ansar et al., 2002; Nakato, 2003; Tokyay and Constantinescu, 2005a) due to the importance in determining pump bays performance. Unfortunately, the phenomena are so complex and diverse that, there is no comprehensive theoretical model to predict them. Existing design guides, usually contains little more than rules of thumb for pump performance. Therefore, Computational Fluid Dynamics (CFD) could play a potentially useful role in determined flow conditions within pump bays. Improved knowledge of flow conditions should lead to improvements in the bay design and consequently the pumps operation. The factors affecting pump-bay flows have been known in qualitative and empirical terms but there is no exact method for predicting them. The only way of doing so is by an expensive hydraulic model study. Because of the high costs involved in the design and construction of small physical scale laboratory models, there is a need for more research in the numerical simulations. The prime drawbacks of physical hydraulic models are the relatively lengthy periods needed for model building, data acquisition and analysis. The numerical simulation code introduced herein does not have those drawbacks, plus it has additional advantages. In this study, the application of the comprehensive CFD model will be used in the design of pump bays, and the comparison with the development experimental model will be discussed in details.

1.2 Problem Statements

Water pumps in drainage, agriculture, and industrial process applications are known to experience certain common operational problems, such as vibration, impeller damage due to cavitation and excessive bearing wear resulting in severe deterioration of their performance and finally lead to a significant increase of operational and maintenance costs. These problems probably result from poor intake design. Therefore, there are several problems that should be highlighted in this study. The common of the main problems are summarized as follows (Larsen and Padmanabhan, 2001; Karassik et al., 2001 and Nakato, 2003):

a. Free surface vortices – the air may draw from the surface into the pump. These types of vortices can cause unbalance loading of impeller, periodic vibration and therefore reduction in pump capacity.
b. Subsurface vortices – which may emanate from floor, site, back walls or combination among them. These can cause vibration and cavitation that may reduce pump efficiency.
c. Pre-rotation – flow entering the pump which change the angle of the attack of the impeller blades from the design value and may effect pump efficiency and lead to cavitation.
d. An uneven distribution of flow at the pump throat which may results in unequal loading of pump impeller. This action will lead to vibration and unbalance loading of impeller.
e. Cavitation that can cause damage on the underside of mixed flow impeller.

These problems encountered in the pump sump will affect the pump performance and significantly increase the operational and maintenance costs. In order to identify sources of particular problems and find practical solution for it, the usual approach is to conduct the laboratory experiments on a scaled physical model. The problems are already there; however the solutions are still the matter. The main concern and challenge in this study is to understand the flow characteristics. These results will be compared with the computational result by using the Computational Fluid Dynamic (CFD), FLUENT^{TM} software.

1.3 Objectives of the Study

In order to accommodate the main concern and challenge stated in the problem statement, the following objectives are set up to be find the optimum solution. These designated objectives will serve as the basis of the problem solving and also as a guideline and reference in order to complete the study. The objectives are as listed below:

1. To identify flow characteristics in particular surface vortices and subsurface vortices, velocity distribution and pressure contour at different discharge water levels.
2. To develop a simulation model of sump intake using CFD.
3. To ascertain the effectiveness of the CFD analysis.
4. To predict the occurrences of vortices using CFD.

1.4 Scope of the Research

To achieve the above objectives designed, the following tasks need to be carried out:

(i) Literature review – This is the first step to understand the concept and theory of the related research area. It serves as a basic knowledge of other researchers experiences as stated in their literature works reviewing the theory, concept and methods of their studies. It is hoped that the review will ensure a good overview on the whole research activities.

(ii) Design of the structure of physical model – The structure of the physical model had been designed by referring to the existing design guidelines (DID, 2000). The design of the pump sump includes the determination of material used, sump scale and relevant components such as the approaching slope and bell mouth.

(iii) Construction of the physical model – In this part, experiences and expertise from Drainage and Irrigation Department (DID) was utilized, by involving in construction of a physical model. Construction of the physical model includes column intake, inlet channel, water intake, piping system, fitting and checking valve and pumps.

(iv) Hydraulic data collection on the model testing – In this study, three types of measurement were conducted which involve velocity, flow and swirl angle measurements. These three measurements were conducted using different special equipments that will be explained in the following chapter. Visual tests that engage the dye tracing technique were also carried out to understand and identify the flow characteristics and vortex position.

(v) Numerical simulation – The simulation had been made by using CFD code i.e. FLUENT^{TM} 6.2 software. The FLUENT^{TM} model serves as a tool that discrete and solves governing equations for specific geometries using a set of finite volume method.

(vi) Data processing, analysis, interpretation and evaluation – At this stage, all the data from experimental methods and numerical methods have to be analyzed. Both experimental and simulated data will be interpreted and evaluated accordingly.

(vii) Result assessment – This section presented results and analysis of data from experimental physical model test, simulated by using CFD method and its comparison.

(viii) Conclusions and recommendation – It is the final chapter for this thesis, which highlights the findings of this research and recommendations for further studies on the related topic.

1.5 Advantages of the Research

The outcomes of the research will provide some advantages in understanding the flow feature by experimental and numerical methods. The entire advantages are as listed below:

1. The vortices including surface vortices and subsurface vortices can be locally identified in the numerical model. Perhaps, other hydraulic problem such as back flow and dead flow region can also be identified.
2. By using numerical method, in stead of construction the physical model of pump sump, the numerical results could be utilized. These could reduce cost and working time.
3. Operational and maintenance cost of a pump station can be cut off by curbing the entire hydraulic problem through more efficient and effective design.
4. The developed databases can be used to remedy existing problematic pump sump which can help other researchers.

1.6 Thesis Structure

The thesis has been categorized into specific chapters for better understanding of the research. The lists of chapters are as follow:

Chapter 1: Introduction – This chapter gives an overview of the thesis including five important things such as background of the study, problem definition, objective of the research, scope of research and advantages of the research.

Chapter 2: Literature review and hypothesis – This chapter provides important theoretical and conceptual understanding of related topics based on various researches including hypothesis of the research.

Chapter 3: Experimental setup – The experimental setup of the sump intake model will be described and it will help to fulfill the proposed designated objectives and answer the problems defined. This chapter is the most important part in this thesis. The experimental procedures include the data collection procedure using instrumentations.

Chapter 4: Computational Fluid Dynamics – This chapter will discuss the concept, theory and the methods of CFD. Besides that, this chapter will give a clear view and step by step basic understanding of CFD.

Chapter 5: Result and discussion – Results, analysis, discussion and result assessment of experimental and simulation are described in this chapter. Comparison of experimental and numerical methods results are described in detail. It will be followed by analysis, discussion and result assessments. Furthermore, the result from the CFD methods will be visualized and discussed in this chapter.

Chapter 6: Concluding remarks – The final chapter will summarize all the activities related to this study, and all the recommendations for further works are presented here.

CHAPTER 2
LITERATURE REVIEW

2.1 Introduction

This chapter describes the pump intake structure, Reynolds number, Froude number, similitude analysis, mechanism involved in vortex formation, free surface and water interface, fundamental of vortex flow in sumps and problem encountered in the pump intakes. Finally, a review is presented on Computational Fluid Dynamics (CFD) and their application in the physical model of a pump sump.

2.2 Pump Intake Structures

The term 'water intake' refers to the channel leading from the water source which may be a river or reservoir and all installations downstream including the pump column or the intake pipe (the suction tube portion of a vertical intake), the approach channel (upstream of pump bay) and the pump bay (bounded by the floor, the back wall, side walls dividers walls separating adjacent pump column). Usually the upstream end of the pump column has an inlet attachment called the suction bell. The function of the intake is to supply an evenly distributed flow of water to the pump suction bell.

Intake structures can be categorized as being clear liquids or solids-bearing liquids. For clear liquids, intakes are further classified into rectangular, formed, circular and trench types, as well as suction tanks and cans. For solid-bearing liquids, trench type and rectangular wet wells are usually considered. These structures are covered in detailed laboratory studies where hydraulic modeling is frequently performed to locate and suppress or avoid flow problems in existing water intakes. For example, Larsen and Padmanabhan (2001) recommended model study to be undertaken for the following water intakes conditions:

- Intakes with asymmetric approach flow (e.g., an offset in the approach channel)

- Intakes with multiple-pump sumps with a common approach channel and variety of pump-operating combinations.
- Intakes with pumps capacities greater than 2.5m^3/s per pump.
- Intakes with an expanding approach channel and
- Intakes with possibilities of screen blockages and/or obstructions close to the suction-pipe entrance.

A hydraulic intake structure, such as the multiple pump sump, consists of an open channel (pump sump or diversion channel) and a pipe or conduit. The flow in the intake involves the transition from a free-surface flow in an open channel to a close conduit flow in a pipe. Several of types of intake structure exist. Figure 2.1 shows the different types of intake structures. The vertically downwards intake consists of a pipe or conduit located just above (or near) the floor of the pump sump. Other intake structures include a horizontal intake, inclined downward and upward intakes, and vertically upward intakes in free-surface flow. If the flow is not driven by gravity, as in the vertically and inclined downward intakes, a pipe is needed to withdraw water from the pump sump to its final destination. Therefore, a pump is required in the horizontal, the vertically upward and vertically inclined intakes. The intake that requires pumps are commonly referred to as pump intakes. For this study, vertically upward intake is used in the physical model of pump sump.

There are many researchers who have stated about the specific hydraulic phenomena that can adversely affect the performance of pumps (Tullis, 1979; Dicmas, 1987; Bauer and Nakato, 1997; Larsen and Padmanabhan, 2001; ANSI, 1998 and Warring, 1984). The hydraulic phenomenon that has been discussed are free surface and subsurface vortices, excessive of flow entraining the pump and its variations with time, entraining air or gas bubbles and non-uniform of velocity at the impeller eye and excessive variations in velocity with time.

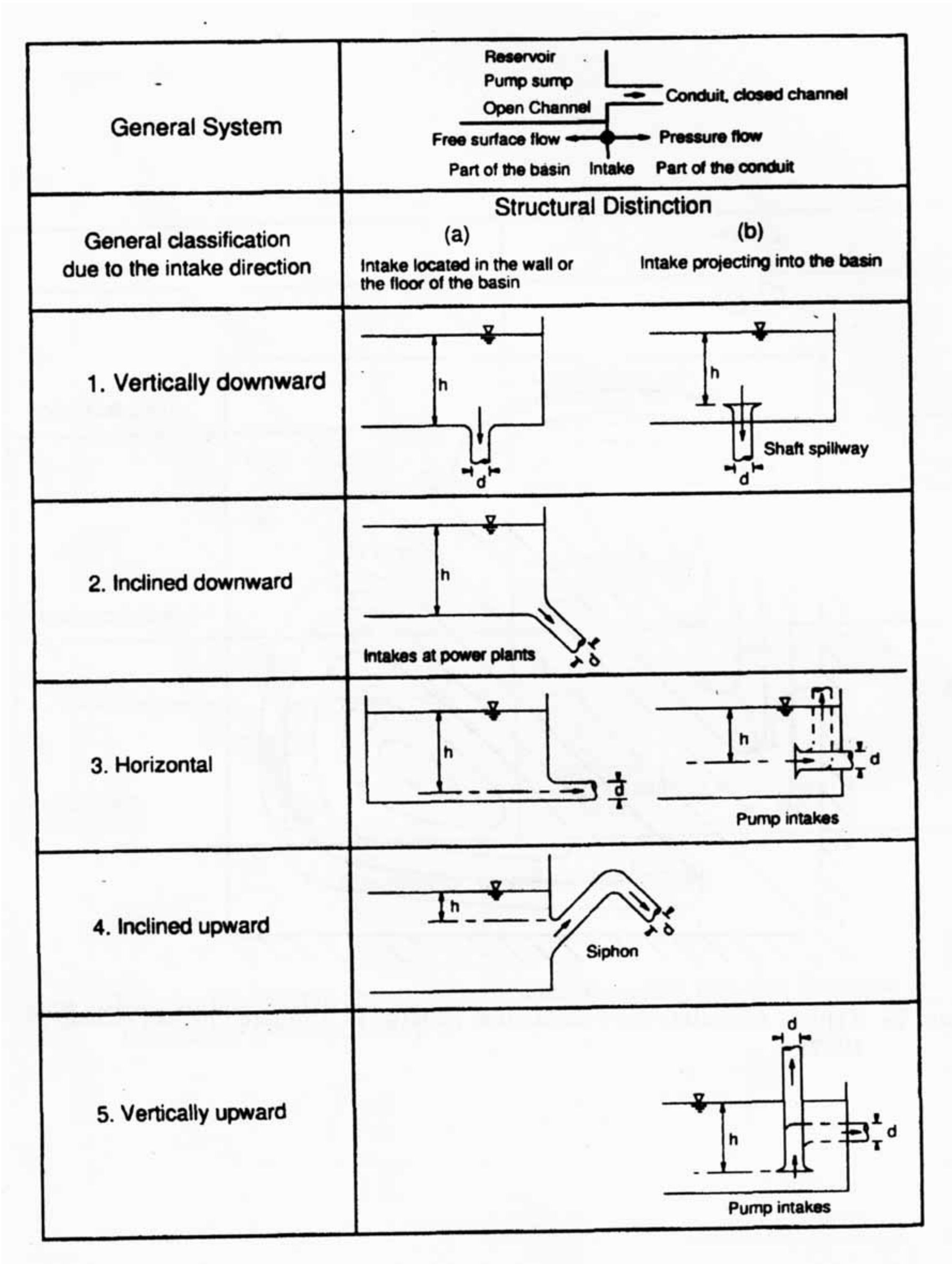

Figure 2.1: Types of intake structures (Source: Knauss, 1987)

The sump should be designed to allow the pumps to achieve optimum hydraulic performance for all operating conditions. The acceptance criteria for the model test are based on the Hydraulic Institute Standards (1983) recommendations and shall be as follows:

- Free surface and subsurface vortices entering the pump must be less severe than vortices with coherent (dye) cores (free surface vortices of Type 3 and subsurface vortices of Type 2)
- Dye core vortices may be acceptable only if they occur for less than 10% of the time or only infrequent pump operating conditions.
- Swirl angles, both the short-term (10 to 30 second model) maximum and the long-term (10 minute model) average indicated by the swirl meter rotation, must be less than 5°. Odgaard and Dlubac (1984) reported swirl rotation must be less than 3°.
- Maximum short-term (10 to 30 second models) swirl angles up to 7 degrees may be acceptable, only if they occur less than 10% of the time for infrequent pump operating conditions. The swirl meter rotation should be reasonably steady, with no rapid changes in direction when rotating near the maximum allowable rate (angle).
- Time-average velocities at points in the throat of the bell or at the pump suction in a piping system shall be within 10% of the cross-sectional area average velocity. Time-varying fluctuations at a point shall produce a standard deviation from the time-averaged signal of less than 10%.

A set of design criteria has been developed by the Iowa Institute of Hydraulic Research (IIHR) based on their vast experience with model studies of pump sumps. Nakato and Yoon (1992) summarize them as follows:

- No detectable boundary-attached vortices extending into the pump bells

- No free-surface vortices stronger than Type 2 (Arden Research Laboratory classification)
- No velocities measured at the pump throat that vary by more than 10% from the average of all local velocities measured in the cross section
- Vortimeter-tip velocity angles (swirl angles) no greater than 5°
- No detectable, large scale, persistent unsteadiness or waviness in the pump bell approach flows, no indication of persistent large scale turbulence, and no flow anomalies judged objectionable by investigators experienced with pump-intake model test.

The guidelines listed could help to determine whether conditions for the existing intake structures are acceptable or not. If the conditions are not acceptable, modification to the intake structure should be made until the requirement is satisfied. There are many guidelines or the basic designs that have been developed to improve the reliability and performance of pump sumps. The Hydraulic Institute Standards (1983) and British Hydrodynamics Research Association (Prosser, 1977) detailed some recommendations for the multiple pumps in open sumps as illustrated in Figure 2.2 and Figure 2.3.

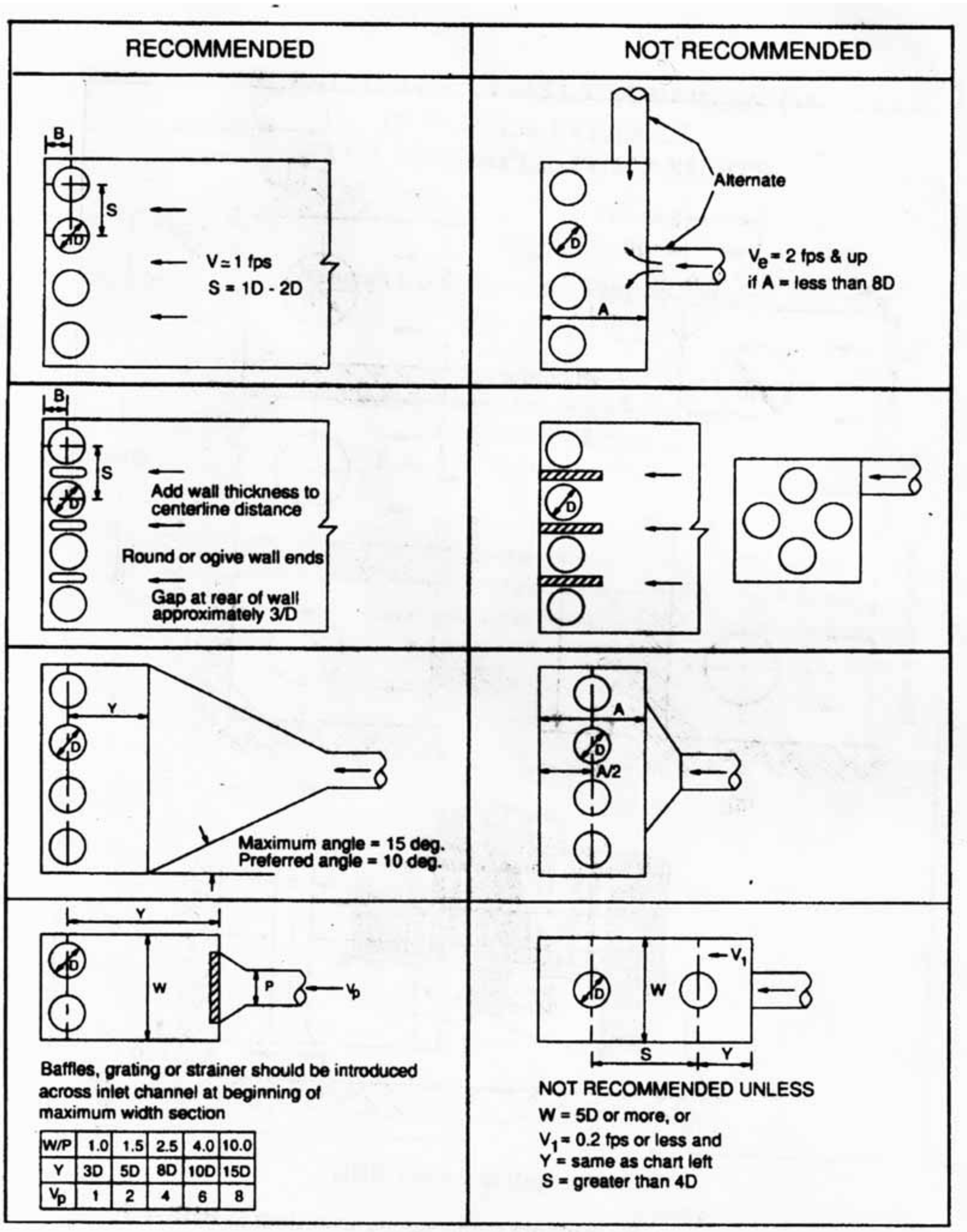

Figure 2.2: Guidelines for multiple pump sumps (Source: Hydraulic Institute Standards, 1983)

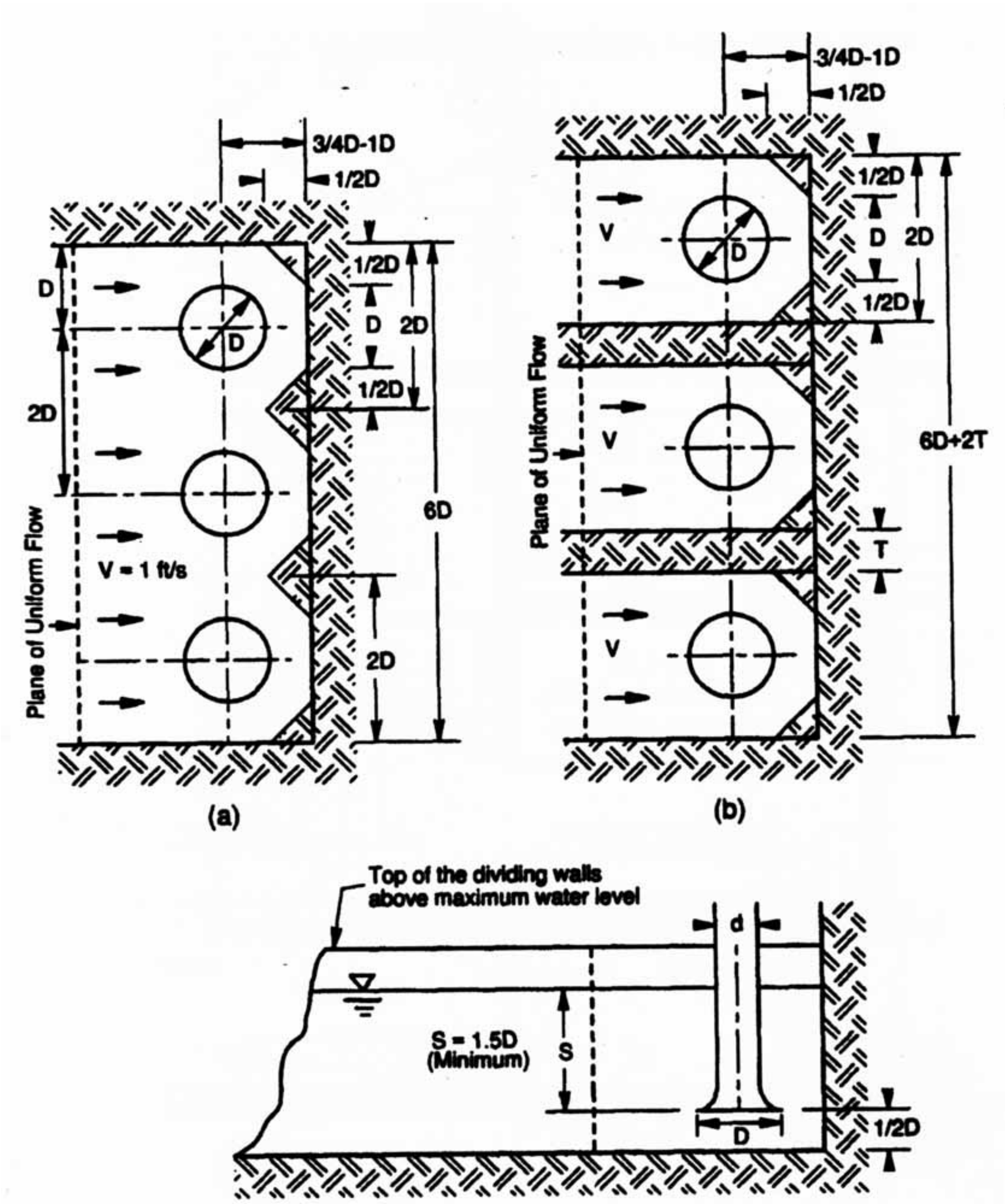

Figure 2.3: Basic sump design for multiple pump sumps according to British Hydrodynamics Research Association, (a) open sump and (b) unitized sumps (Source: Prosser, 1977)

Padmanabhan (1987) has also developed guidelines for single sump and multiple sump intakes, which are illustrated in Figure 2.4 and Figure 2.5 These guidelines and basic sump design as discussed are considered in this study in determining methods to minimize the vortices in the multiple pump sumps. As Padmanabhan (1987) states, the guidelines given previously are helpful for the preliminary design of pump sumps, but a model study should be performed for more complex intake structures and for the evaluation of preliminary designs.

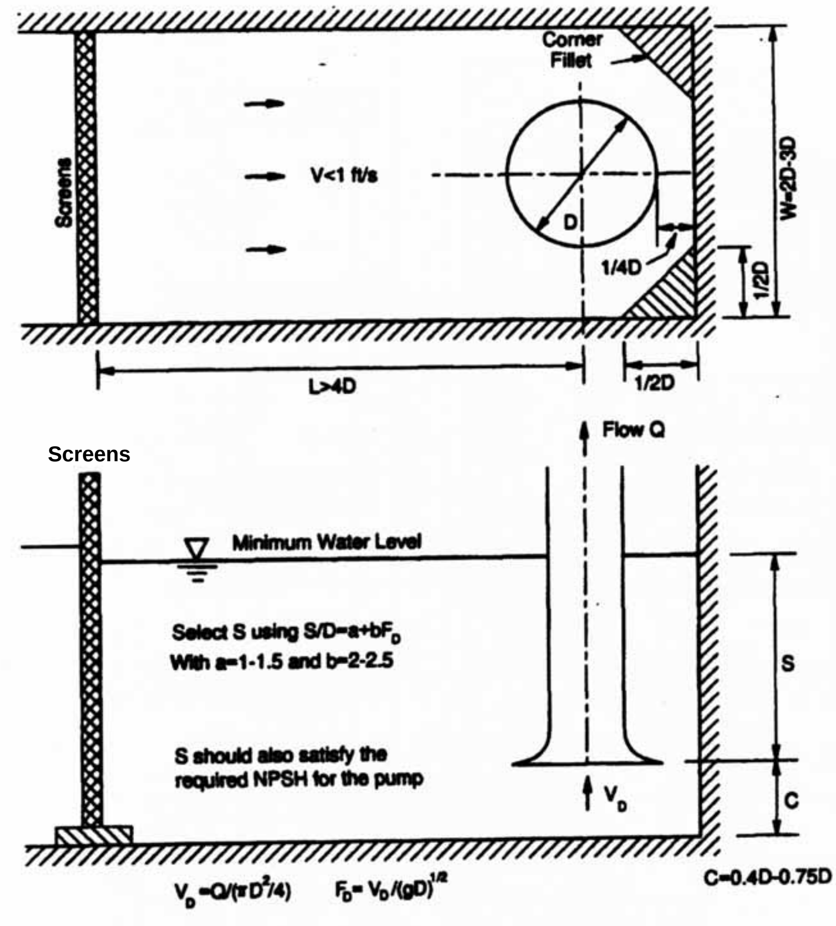

Figure 2.4: Basic design for a single bay sump with uniform approach flow (Source: Padmanabhan, 1987)

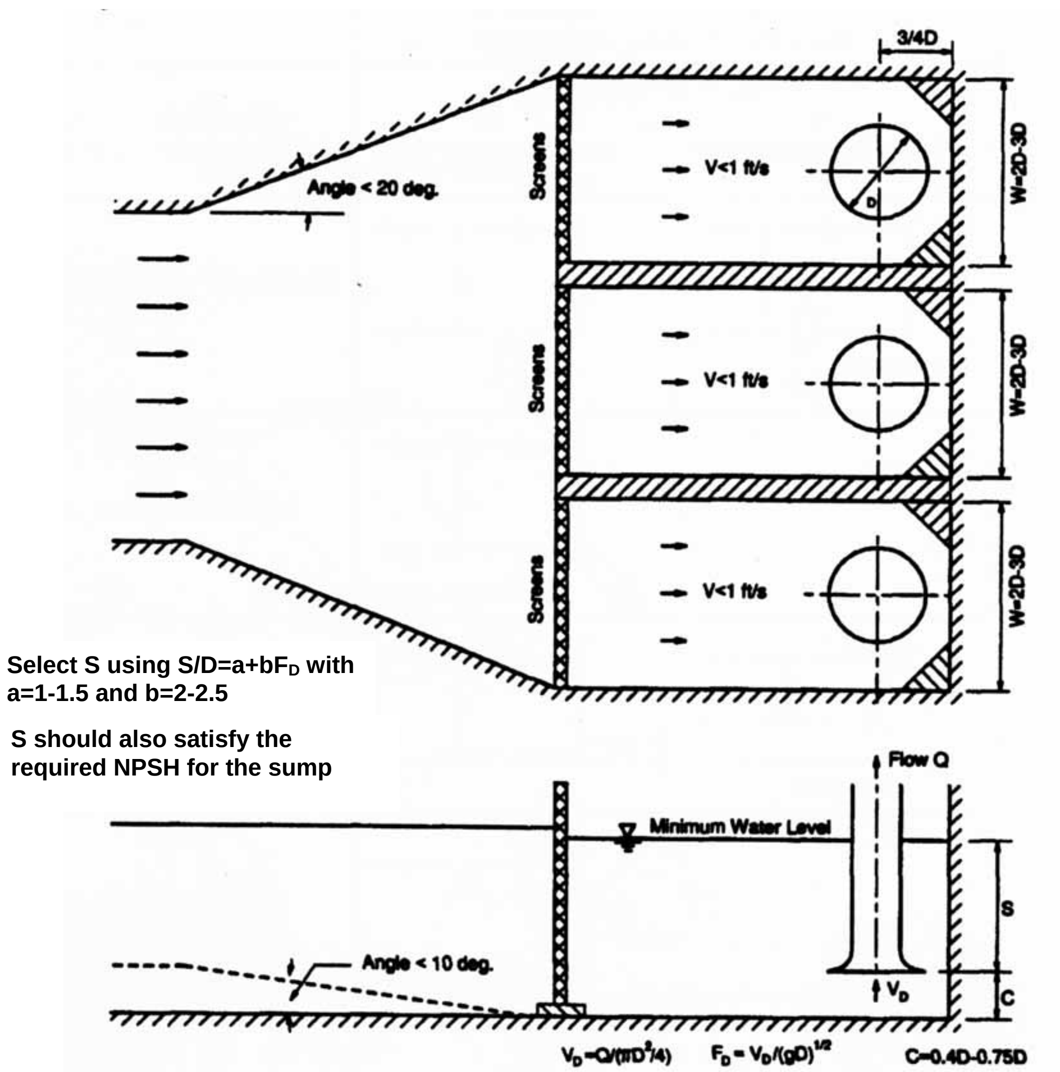

Figure 2.5: Basic design for multiple bay sumps with uniform approach flow (Source: Padmanabhan, 1987)

The American National Standard for Pump Intake Design (1998) recommends that the intake design for vertical wet pit pumps is as shown in Figure 2.6 and Figure 2.7. The geometry is generally defined in terms of the pump inlet bell diameter as shown. Once the number and the size of pump required are determined, a pump inlet diameter can be estimated. The bell diameter can be estimated based on an inlet pipe velocity of between 0.9m/s and 2.4m/s. As the selected bell diameter has been determined, the proportions of the inlet structure can be estimated from Figure 2.6 and Figure 2.7. Table 2.1 gives recommended values for the dimensions.

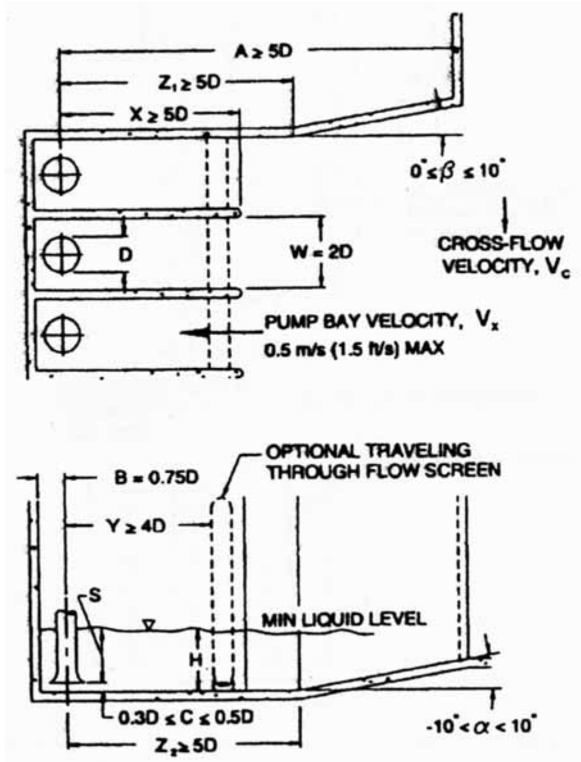

Figure 2.6: Recommended intake structure layout (Source: American National Standard for Pump Intake Design, 1998)

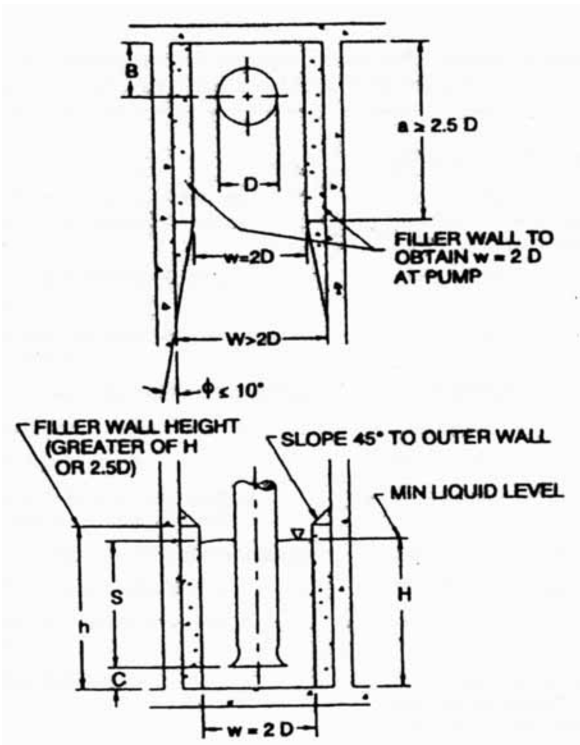

Figure 2.7: Filler wall details for proper bay width (Source: American National Standard for Pump Intake Design, 1998)

Table 2.1: Recommended dimensions for Figures 2.6 and 2.7 (Source: American National Standard for Pump Intake Design, 1998)

Dimension Variable	**Description**	**Recommended Value**
A	Distance from the pump inlet bell centerline to the intake structure entrance	A = 5D minimum, assuming no significant cross-flow* at the entrance of the intake structure
a	Length of constricted bay section near the pump inlet	a = 2.5D minimum
B	Distance from the back wall to the pump inlet bell centerline	B = 0.75D
C	Distance between the inlet bell and floor	C = 0.3D to 0.5D
D	Inlet bell design outside diameter	(see text)
H	Minimum liquid depth	H = S + C
h	Minimum height of constricted bay section near the pump inlet bell	h = (greater of H or 2.5D)
S	Minimum pump inlet bell submergence	$S = D\ (1.0 + 2.3F_D)$
W	Pump inlet bay entrance width	W = 2D minimum
w	Constricted bay width near the pump inlet bell	w = 2D
X	Pump inlet bay length	X = 5D minimum, assuming no significant cross-flow at the entrance to the intake structure
Y	Distance from pump inlet bell centerline to the through-flow traveling screen	Y = 4D minimum. Dual-flow screens required a model study
Z_1	Distance from pump inlet bell centerline to diverging walls	Z_1 = 6D minimum, assuming no significant cross-flow* at the entrance to the intake structure
Z_2	Distance from pump inlet bell centerline to sloping area	Z_2 = 5D minimum
α	Angle of floor scope	α = -10 to + 10 degrees
β	Angle of wall convergence	β = 0 to +10 degrees(Negative values of β, if used, require flow distribution devices developed through a physical model study)
Φ	Angle of convergence from constricted area to bay walls	Φ = 10 degrees maximum

*Cross-flow is considered significant when $V_c > 0.5V_x$ average

2.3 Reynolds's Number

In 1883, Osborne Reynolds is the first to develop the basic laws of turbulent flow. He studied the flow of liquid in pipes and found that at a low speed the flow is smooth but at high speed, the flow is turbulent. He found that the onset of turbulence in a smooth pipe was related to the Reynolds number in a very interesting way. In the case of pipe flow, where the diameter of the pipe is the characteristic length scale, for Re $\leq$ 2000 the flow is laminar, for 2000 < Re < 4000 there is a gradual change to turbulent flow, and for Re > 4000 the flow is turbulent. In the case of open-channel flow, such as in canals and rivers, the depth of flow is used as the characteristic length scale and open-channel flows are turbulent for $\geq$ 1000 (Chin, 2000). In most engineering applications involving closed-conduit and open-channel flow, the Reynolds number limits are far exceeded and the flows are fully turbulent. The formula of Reynolds number can be calculated by the equation:

$$\mathrm{Re} = \frac{\mathrm{VL}}{\upsilon} = \frac{\rho \mathrm{VL}}{\mu} = \frac{\text{inertial forces}}{\text{viscous forces}} \tag{2.1}$$

where,

V = velocity (m/s)
L = characteristic length (m)
υ = kinematic viscosity (m^2/s)
μ = dynamic viscosity (kg/ms)
ρ = water density (kg/m^3)

2.4 Froude Number

The Froude number is the ratio of inertial to gravitational forces and is defined for pump sump as:

$$F_D = \frac{V}{\sqrt{gL}} \tag{2.2}$$

where,

F_D = Froude Number

g = gravitational constant (9.81m/s)

Characteristic length in an open channel is taken to be the hydraulic depth. Depending on the magnitude of the ratio of inertial to gravity forces, a flow is classified as subcritical, critical or supercritical (French, 1986). If $F_D = 1$, the flow is in a critical state with the inertial and gravitational forces in equilibrium. If $F_D < 1$, the flow is in a subcritical state, and the gravitational forces are dominant. If $F_D > 1$, the flow is in a supercritical state and inertial forces are dominant.

When the flow is subcritical, $F_D < 1$, the velocity of flow is less than the speed of an elementary gravity wave. Therefore, such a wave can transmit upstream against the flow, and upstream areas are in hydraulic communication with the downstream areas. Furthermore, when the flow is supercritical, $F_D > 1$, the velocity of the flow is greater than the speed of the elementary gravity wave. Therefore, such wave cannot transmit upstream against the flow, and the upstream areas of the channel are not in hydraulic communication with the downstream areas. Thus, the possibility of an elementary wave transmit upstream against the flow can be used as a criterion for differentiating between subcritical and supercritical flows. Critical flow is unstable and often sets up standing waves between super and subcritical flow. When the actual depth is below the critical depth, it is called supercritical because it is in a higher energy state. Likewise, if actual depth is above critical depth it is called subcritical because it is in a lower energy state.

2.5 Similitude Analysis

In the similitude analysis, the geometric and flow similarity requires the model and the flow to be identical as the real model and flow. This is to ensure the result obtained from the model study is well presented in order to predict full scale behavior. If similarity has been obtained between model and prototype, the Froude number of the model and the prototype must be the same for flow conditions where inertial and gravitational forces are dominant.

For similarity of the flow patterns, the Froude number shall be equal in model and prototype (American National Standards for Pump Intake Design, ANSI/Hydraulic Institute Standards 9.8, 1998):

$$F_r = \frac{F_m}{F_p} = 1 \qquad (2.3)$$

where,

F_r = Froude ratio parameter

F_m = Froude model parameter

F_p = Froude prototype parameter

A reasonable large geometric scale is selected to minimize viscous and surface tension scale effects, and to reproduce the flow pattern in the vicinity of the intake. The model also shall be large enough to allow visual observations of flow patterns, accurate measurements of swirl and velocity distribution and sufficient dimensional control. Froude number and Reynolds number for the model and prototype cannot be made equal. Fixing the same Froude number for model and prototype results in the velocity being reduced in the model depth and a fixed gravity constant. Fixing the Reynolds number results in the velocity being increased in the model, given geometric scale reduction of dimensions and constant kinematics viscosity.

Froude and Reynolds number equality could only be achieved simultaneously by using a fluid with suitable kinematics viscosity in the model to adjust the Reynolds number to match the prototype Reynolds number. However this is not possible and in practice, water is used in the model as well as in the prototype. In the free surface work, the gravitational forces are the most important value, so that the Froude number must be made equal in the prototype in preference to the Reynolds number. This ensures surface profiles, rotational flow and waves are correctly represented. For the turbulent flow, the Reynolds number is not particularly important as long as both model and prototype have values in the same flow regime (Abustan et al. 2004). If the reduced Reynolds number of a model approaches the transitional point of turbulent to laminar flow then laminar flow could occur in the model but turbulent or transitional flow would occur in the prototype. Clearly this is not acceptable and consequently a minimum operable Reynolds number has to be chosen (Abustan et al. 2004).

2.6 Mechanisms Involved in Vortex Formation

A number focused experimental and numerical investigations have provided insight into the fundamental processes leading to the development of vortices in the sump intakes. Shin et al. (1986) demonstrated that two basic mechanisms lead to inlet vortex formation. The first mechanism involves the development of an inlet vortex due to the amplification of ambient vorticity in the approach flow as vortex lines are convicted into the inlet. The second mechanism involves the development of a trailing vortex in the vicinity of the intake as a result of the variation in circulation along the inlet. For this second case, a vortex can develop in a flow that is irrotational upstream, and the vortex development therefore does not depend on the presence of ambient vorticity. Shin et al. (1986) investigation on kinematic parameters, indicate that the strength of an inlet-vortex or trailing vortex system increases with decreasing distance from the surface. However, for an inlet in an upstream irrotational flow, two counter rotating vortices can still trail from the rear of the inlet. Causes of vortex motion, however, are still difficult to define for most practical situations.

2.7 Free Surface and Water Interface

Vortices in the vicinity of pump intakes may be adjacent to the channel bottom or a channel wall (submerged vortices) or they may appear adjacent to the free surface (free surface vortex). By studying the way that the vortex interact with the free surface, more information can be known about the characteristics in vortex formation and possibly better prediction methods can be developed as a result of such studies.

The study of the interactions with the free surface requires consideration of the dynamics of the vorticity field bounded by a deformable surface. The surface deforms to satisfy the conditions that the tangential stress is equal to zero and the normal stress is equal to a constant at all times. According to Rood and Edwin (1995), the interaction between vorticity and the surface characteristized by both the vorticity and flux of the vorticity at the free surface, the deformation of the free surface, and the dynamic behavior of the velocity field. The stresses at the interface between the two fluids (water and air) must be in balance, such that:

$$(\text{stress})_{\text{water side of interface}} + (\text{stress})_{\text{air side}} + (\text{stress})_{\text{surface}} = 0 \qquad (2.4)$$

When describing the local details of flow at the free surface, viscous forces normally cannot be neglected, especially when describing the vorticity generated by the deformation. However, when localized flow details are not taken into account, there are instances when the viscous terms can be neglected. By neglecting viscous forces, and order of magnitude, estimates of the deformations can be obtained. For estimation of the free surface deformation, the approximation is appropriate when the viscous force is much less than the inertial force. This condition typically occurs when the flow is at high Reynolds number.

2.8 Fundamentals of Vortex Flow in Sumps

Deviations in the pump approach flow distribution are the most common source of swirl and vortex formation. Durgin and Hecker (1978) categorized the sources of vortex formation into three types:

- Nonuniform approach flow to the sump due to the geometric orientation of sump or approach channel or due to streaming flow patterns generated by obstructions such as intake piers or columns.
- Existence of shear layers of high velocity gradients, including separated boundary layers which are inherently rotational.
- Rotational wakes generated by objects or obstructions in the way of the approach flow to the sump.

Padmanabhan (1987) reported that items first and second listed above are major sources of vorticity in free surface and submerged vortices in pump sumps, respectively. Similar, Anwar (1968) had mention that, the formation of vortices is governed by two major factors which are submergence depth or distance from the water surface to the entrance of the suction pipe and the swirl in the approach flow. He also stated that a reduction in the swirl or an increase in submergence depth can prevent the formation of vortices. Chang (1977) summarized the flow processes that generate vortices in a pump sump as follow:

- Asymmetric approach flow – the approach flow has an inherent swirl which can be magnified as the flow converges into the intake, due to the conservation of angular momentum.
- Boundary discontinuities – changes in channel cross-section, diffusers, false baffles, etc., can cause small eddies to shed, thus adding to the total vorticity. The importance of this effect depends on how close the discontinuity is to the

intake and the strength of the eddy it creates. The presence of the intake itself can also be a source of shading eddies.

- Boundary layer development – since the velocity at any solid boundary must be zero because of viscosity, velocity gradients will be present in the boundary layer which generate vorticity.
- Stagnation point flow – The flow at the plane surface near a gas turbine intake showed the existence of a stagnation point towards which the boundary layer, containing vorticity, flowed and was subsequently, drawn into the intake. Using a combined boundary layer and potential flow analyses, a concentrating stagnation point in the free surface is associated with vortex formation at hydraulic intakes; and,
- Secondary layer - The presence of secondary flow currents in a plane perpendicular to the main flow direction in straight rectangular channel. These secondary flow velocities are only about 1% of the main velocity, but may be of sufficient strength to contribute to the instability of vortices.

2.9 Problems Encountered in Pump Intakes

The various hydraulic problems associated with pump intakes include formation of surface and subsurface vortices, prerotation and swirl, and flow separation at or near the suction bell of either wet pit or dry-pit centrifugal pump. Any of these problems adversely affect pump performance by causing cavitation, vibrations, and/or loss of efficiency (Tullis, 1979; Alboleda and El-Fadel, 1996). Usually there is more than a single reason for these problems, and the extents of the combined effects are seldom predictable by mathematical modeling.

Formation of vortices, dependent on suction pipe velocity and submergence, is strongly influenced by added circulation from vorticity sources, such as a nonuniform approach flow resulting from intake and approach channel geometries; rotational wakes shed from obstructions, such a columns or piers; and the velocity gradients resulting from boundary layers at the walls and floor (Durgin and Hecker, 1978). The circulation contributed by these vorticity sources is unpredictable and strongly depends on intake design and operating conditions, especially for large pumping units with multiple bays fed by a common approach channel. In these cases, physical modeling is the only way of predicting the behavior of the prototype with a reasonable degree of reliability.

2.9.1 Free Surface Vortex

Jain et al. (1978) reported that air entraining vortices, known as free surface vortices, draw air into the intake from the water surface and thereby cause considerable loss of pump efficiency and produce vibrations and noise. Free surface vortices are most damaging to the pump when they draw air or trash into the intake columns. The intensity of surface vortex varies directly with some functions of intake and/or approach velocity and inversely with submergence, assuming upstream conditions and other effects to be constant. The change in intensity is gradual, so the specific point at which a vortex does or does not exist becomes a matter of definition (Dicmas, 1987). To determine minimum submergence of the outlet pipe in the tank, the Hydraulic Institute (1998) recommended the following relationship:

$$S = (1.0 + 2.3F)\, D \tag{2.5}$$

where,

F_D = Froude number

D = Diameter of inlet opening (m)

S = Submergence (m)

At certain stages of development, formations of vortices can be intermittent, so practical definitions have to allow for these variations. Vortices in a model study usually